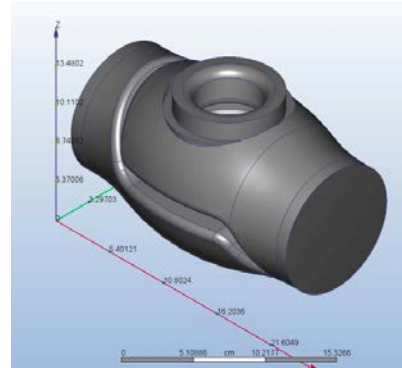


Lesson: Preparing for CFD

In this lesson, you are introduced to different types of boundary conditions that can be used to model fluid motion. You will begin by using Fusion 360 to develop a fluid volume that is transferred to the CFD software, and then add the boundary conditions on the fluid.

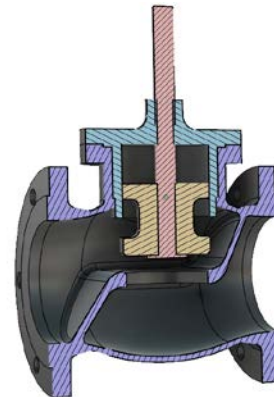
Learning Objectives:

- Simplify the model to prepare it for export.
- Develop a solid model of the fluid volume.
- Export the model directly to Autodesk CFD.



The completed exercise

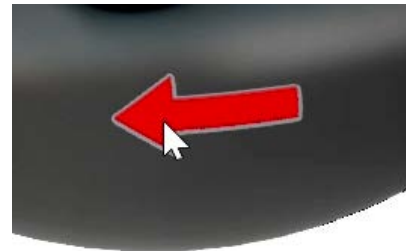
1. Open the file Valve.f3d and save it to your current project. The Valve imports with 4 components. Use the Sectional Analysis to analyze the path of the fluid through the valve.



2. Switch to the Simulation workspace and in the New Study control panel, click Simplify geometry for use in Simulation then click Simplify Model. Click Modify>Remove Features then box select all of the bodies and show only the holes. Delete these holes.



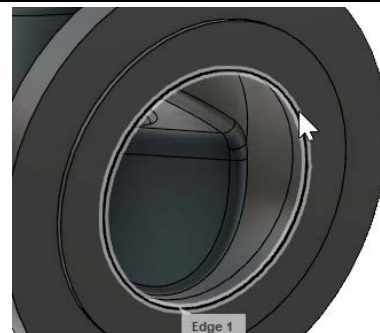
3. Use the Manual features and select the directional arrow; Delete this.



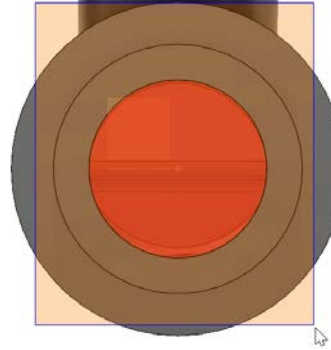
4. To expedite the creation of the fluid solid, combine all the bodies of the Valve. Click Modify>Combine and select the main part of the valve body. Box select the other 3 components. Join these bodies. This is now a single unified component instead of several individual components.



5. Click Create>Patch and select the perimeter of one of the openings. Then restart the Patch tool and select the perimeter of the other opening.



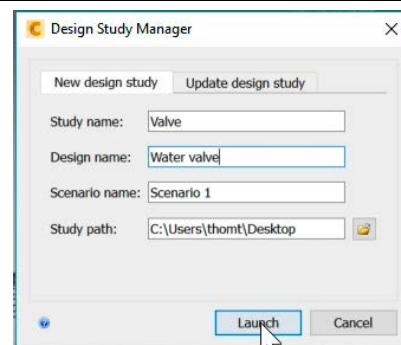
6. Click Create>Fluid Volume. Using a box selection from left to right, select all the internal faces inside the Valve. Click OK. This will create a new component that is the fluid volume.



7. In the Browser, find the new Component5 then right click and select Isolate. You now see the solid model representing the entire fluid volume of the valve as it was positioned at the time of the Join operation.



8. Click Simulation>CFD 2019. Change the design name to Water valve. Select the file path for the study to be created.



9. Examine the options in the Geometry Tools control panel that opens. Close the control panel. You are now ready to prepare it for the simulation study.

